

Available online at www.sciencedirect.com**SciVerse ScienceDirect**

Procedia Engineering 31 (2012) 422 – 427

**Procedia
Engineering**

www.elsevier.com/locate/procedia

International Conference on Advances in Computational Modeling and Simulation

Dynamic Mesh Method for Two-Dimensional Synthetic Jet

Xiaoyong Ma^{a*}, Zhaolin Fan^b, Xingya Da^a^aHigh Speed Aerodynamics Institute, China Aerodynamics Research and Development Center, Mianyang 622661, China^bChina Aerodynamics Research and Development Center, Mianyang Sichuan 621000, China

Abstract

A numerical simulation method using dynamic mesh for two-dimensional synthetic jet flow was developed, in which dynamic mesh conservation equations and spring-based smoothing method were used. A computing model of an isolated jet to validate this method, the max velocity of oscillation is 0.47 m/s, Renault number is 180 and the Strouhal number is 0.056. The simulation results of different phases show that a shear layer is formed at the orifice edge and the shear layer rolls up to form a vortex ring, then fluid nearby the orifice enters into the cavity, and the vortex pair moves away from the orifice under its own momentum when the membrane moves to the bottom position. The suction process is affected by the outside vortex ring, and the second vortex pair impacts acutely the vortex ring outside. Comparing with other similar examples, the results show that the flow structure and action mechanism of synthetic jet flow are reasonable, and therefore this method could be used to simulate two-dimensional synthetic jet flow.

© 2011 Published by Elsevier Ltd. Selection and/or peer-review under responsibility of Kunming University of Science and Technology. Open access under [CC BY-NC-ND license](http://creativecommons.org/licenses/by-nc-nd/3.0/).

Keywords: Dynamic mesh; synthetic jet; unsteady flow; numerical simulation.

Nomenclature

A	amplitude (m)
d	width of the orifice (m)
f	frequency (hz)
L_0	stroke length (m)
Re	Reynolds number

* Corresponding author. Tel.: +86-816-2462264; fax: +86-816-2462259.

E-mail address: mxycardc@mail.ustc.edu.cn.

St	Strouhal number
T	cycle (s, 1/f)
$\vec{u}_0(t)$	centerline velocity (m/s)
φ	angle of phase (°)

1. Introduction

As shown in Fig.1, a typical synthetic jet actuator is a zero net mass flux jet, which consists of an oscillating membrane located at the bottom of a cavity having small orifice in the face opposite the membrane. As the results of sucking and blowing produced by the movement of diaphragm, the net mass flux is zero over one period. As a useful active flow control method, synthetic jet has more advantage such as reducing structural weight, operating costs and emissions. Recently, more experiment and numerical simulation about synthetic jet[1-8] are carrying out to get sufficient understanding of the interaction process and flow characteristic.

Computational fluid dynamic(CFD) is an essential technology in understanding and predicting the flow characteristics of complex flow field, especially for new designs and flow conditions where experimental testing is difficult and expensive. In this note, a numerical simulation method using dynamic grid of CFD for two-dimensional Synthetic jet flow is developed, and numerical simulations are conducted to study synthetic jet flow structure and action mechanism.

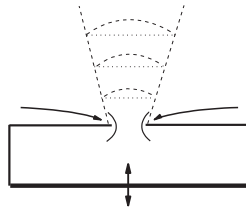


Fig.1. Schematic of a synthetic jet

2. Numerical Method and Computing Model

2.1. Numerical Method

Structured or unstructured meshes offer different advantages for computing model. When geometries are complex, an unstructured mesh can often be created with far fewer cells than the equivalent mesh consisting of structured elements. Farther more, unstructured meshes offer many advantages for moderately-complex geometries. This is because an unstructured mesh allows cells to be clustered in selected regions of the flow domain, whereas structured mesh will generally force cells to be placed in regions where they are not needed.

In this paper, a method named unstructured meshes with spring analogy [9] used to simulate the grids' movement of the computing model. The spring analogy model treated each interior edge of the mesh as a spring with certain stiffness. When any nodes on the boundary move as effect of the external force, the movement will be transferred trough the spring to other nodes, see Figure 2. All the nodes of flow regions would be moving under the action of spring analogy, and the velocity of nodes nearby the moving boundary is more quickly than that of far away from it.

In the numerical programs, the mesh updated is:

$$\vec{x}_i^{t+\Delta t} = \vec{x}_i^t + \frac{\sum_j^n k_{ij} \Delta \vec{x}_j}{\sum_j^n k_{ij}}$$

$$k_{ij} = 1 / \sqrt{|\vec{x}_j - \vec{x}_i|}$$

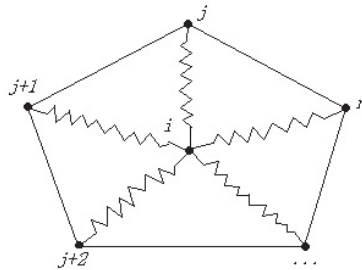


Fig.2. Spring analogy method

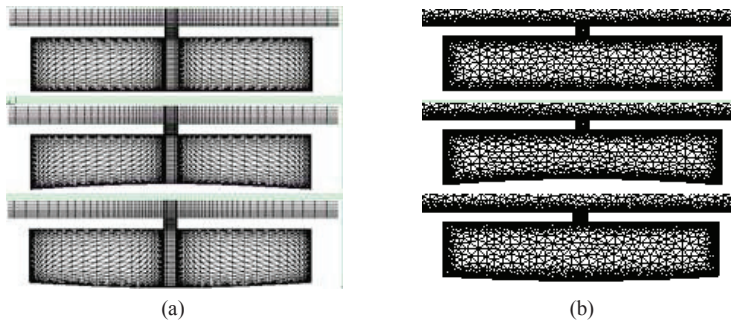


Fig.3. (a) Grids of reference [3]; (b) Present grids

where \vec{x}_i and \vec{x}_j denote respectively the original position of nodes(i and j), k_{ij} is the stiffness of the spring between two nodes(i and j). \vec{x}_i^t and $\vec{x}_i^{t+\Delta t}$ are respectively the position vectors of node(i) at time of t and $t + \Delta t$.

Time-average incompressible Navier-Stokes equation is used to simulate the flow in synthetic jet. The time derivative term is used a first-order implicit scheme, viscosity term is used a first-order backward difference formula, and one equation Spalart-Allmaras turbulence model is chose to compute turbulent flow. All the boundary wall are non-slip and adiabatic, and outfield boundary is dealt with by extrapolation method. When boundary is moving, the integral form of the conservation equation on an arbitrary control volume, V , can be written as

$$\begin{aligned} \frac{d}{dt} \iiint_V \rho \phi dV + \iint_{\partial V} \rho \phi (\vec{u} - \vec{u}_g) \cdot \vec{n} dF \\ = \iint_{\partial V} \Gamma_\phi \nabla \phi \cdot \vec{n} dF + \iiint_V S_\phi dV \end{aligned}$$

Where ϕ is a general scalar, ∂V is used to represent the boundary of the control volume V , dF is infinitesimal area of the control volume's surface, ρ is the fluid density, \vec{u} is the flow velocity vector, \vec{u}_g is the grid velocity of the moving mesh, Γ_ϕ is the diffusion coefficient and S_ϕ is the source term of ϕ .

Table 1. Comparison of computing conditions

	Reference [10]	Present
Domain size	<i>unknown</i>	30 mm × 40 mm
Cavity size	20 mm × 5 mm	20 mm × 5 mm
Orifice width d	1 mm	1 mm
Orifice depth h	1 mm	1 mm
Frequency of oscillation f	150 Hz	150 Hz
Amplitude of diaphragm A	≈ 0.5 mm	0.5 mm
Curve of diaphragm y	<i>unknown</i>	$ax^2 + bx + c$

Where $u_0(t)$ is mean velocity of jet through the orifice, T is the time of oscillation in one cycle, d is orifice width, ρ and μ are respectively density and viscosity of fluid. The equation of oscillation movement is:

$$\vec{u}_b = 4f(y - y_0) \cos(\omega t) \vec{j}$$

where $y = ax^2 + bx + c$, is the curve of diaphragm and $\omega = 2\pi f$ ($f = 1/T$, the frequency of oscillation) is the angular velocity. Also a , b and c were assigned, in this paper:

$$y_0 = 0$$

$$a = -7.854, b = -0.005, c = -0.006$$

Figure 3 shows the dynamic grid movement for the typical synthetic jet geometry of reference [10] and this paper.

2.2. Computing Model

Computing model come from the reference [10], an example of Professor N. Qin, University of Sheffield, UK. Approximations of outfield and curve of diaphragm are used because they are unknown in the paper. Table 1 shows the comparison of computing conditions between reference [10] and this paper. The max velocity of oscillation is 0.47 m/s, the Renault number is 180 and the Strouhal number is 0.056. They are given by equations:

$$Re = \rho U_0 d / \mu, St = df / U_0$$

$$U_0 = L_0 / T, L_0 = \int_0^{T/2} u_0(t) dt,$$

3. Validation and Discussion

Simulation was performed to validate the methods and the behavior of an isolated jet was characterized. As mentioned above, approximations of outfield and curve of diaphragm are used so that the computing results are different. In this paper, vortices magnitude and streamlines pattern are presented (Fig.4 & 5).

Fluid is expelled through the orifice when phase is 90° , a shear layer is formed at the orifice edge and the shear layer rolls up to form a vortex ring, which are similar with the vortices magnitude in Fig.4 (b). When phase is 180° , the membrane moves downwards, fluid nearby the orifice enters into the cavity, and the vortex pair moves away from the orifice under its own momentum. The velocity of the vortex pair moving is smaller than that of reference [10], because the parameters is changed (In this note, the outfield, amplitude and curve of diaphragm are different, $Re=180$ & $St=0.056$). It is evident when phase is 270° , the membrane moves to the bottom position. The suction process is affected by the outside vortex ring, which is not far enough from the cavity (Fig.4 & 5). When phase is 360° , the membrane moves to the original position and fluid spurts out again. The second vortex pair impacts acutely the vortex ring outside, and the fluid field and streamlines pattern (Fig.5 d) are changed obviously. To sum up, the vortices magnitude and streamlines pattern show that the flow structure and action mechanism of synthetic jet are reasonable in the ‘blow’ and ‘suck’ actuation circle. The unstructured meshes method with spring analogy is useful to simulate the behavior of an isolated synthetic jet.

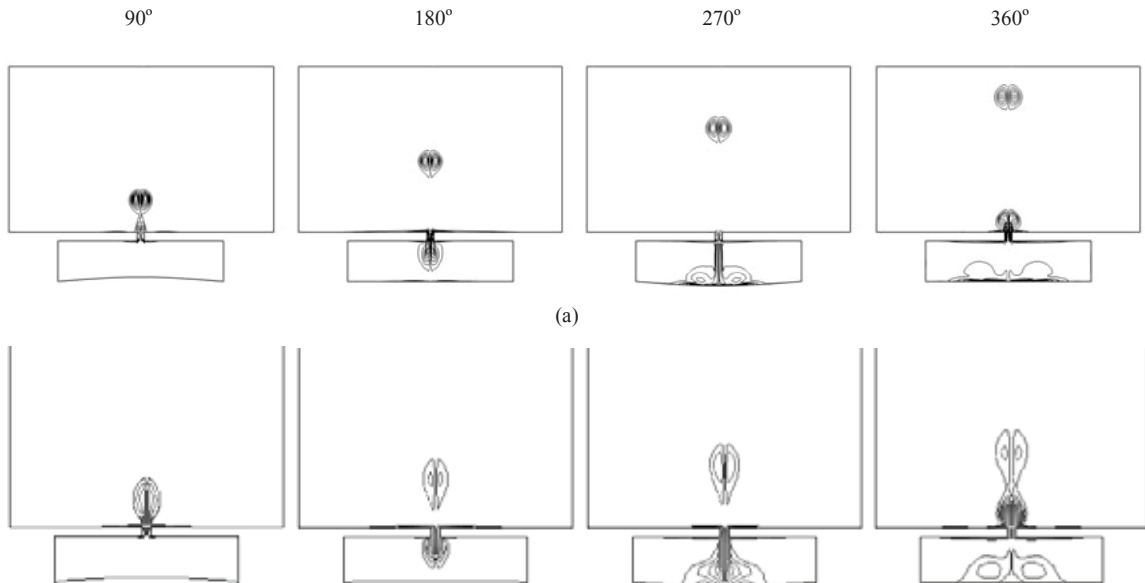


Fig.4. Vortices magnitude of the first cycle (a) Result of reference [10]; (b) Present result

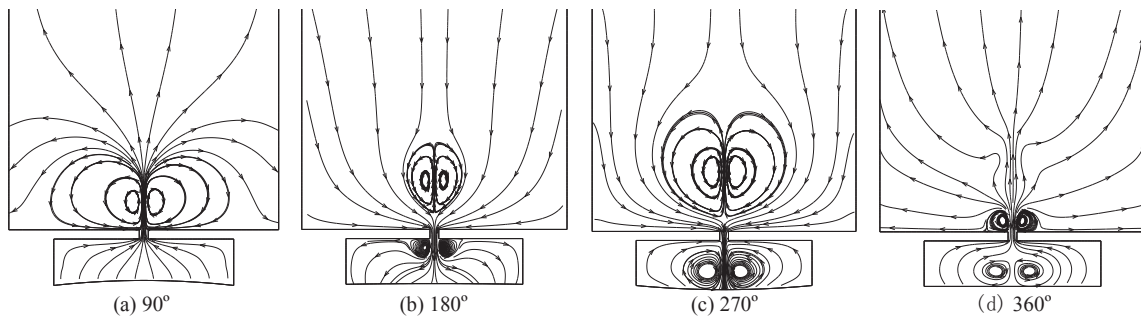


Fig.5. Streamlines pattern of the first cycle

4. Conclusion

In this paper, we introduced a method named unstructured meshes with spring analogy for two-dimensional synthetic jet flow. Numerical simulations of synthetic jet flow are conducted to validate the method with time-average incompressible Navier-Stokes and dynamic grid conservation equations. Compared with other similar examples, the results show that the flow structure and action mechanism of synthetic jet flow are reasonable, and the method could be used to simulate 2D synthetic jet flow.

Acknowledgements

The first author, Xiaoyong Ma, would like to acknowledge the financial supported by International Science and Technology Cooperation Foundation of China under tasks monitored by Dr. Wenhua Wu. All the simulations were performed at CARD-2 Xeon Linux Cluster. Sincere thanks go to Dr. Dangguo Yang for helpful discussion.

References

- [1] P. Catalanoy, M. Wang, G. Iaccarino, Ivo F. Sbalzarini, AND P. Koumoutsakos. Optimization of cylinder flow control via actuators with zero net mass flux. Center for Turbulence Research, Proceedings of the Summer Program 2002: 297-303.
- [2] Quentin Gallas, Jose Mathew, Anurag, Kasyap, and et al. Lumped element modeling of piezoelectric-driven synthetic jet actuators. AIAA Paper 2002-0125, 2002.
- [3] Matteo Pes, Bojan Lukovic, Paul D. Orkwis, Mark G. Turner. Modeling of two dimensional synthetic jet unsteadiness using neural network-based deterministic source terms. AIAA Paper 2002-2860, 2002.
- [4] Quentin Gallas and Ryan Holman. Low dimensional modeling of Zero-Net Mass-Flux actuators. AIAA Paper 2004-2413, 2004.
- [5] SH, Kim and C, Kim. Separation control on NACA23012 using synthetic jet. AIAA Paper 2006-2853; June 2006.
- [6] McCormick DC. Boundary layer separation control with directed synthetic jets. AIAA Paper 2000-0519; January 2000.
- [7] J, Zhou and S, Zhong. Numerical simulation of the interaction of a circular synthetic jet with a boundary layer. Computers & Fluids, 2009, 38: 393-405.
- [8] SH, Kim and C, Kim. Separation control on NACA23012 using synthetic jet. AIAA Paper 2006-2853; June 2006.
- [9] Batina, J. T. Unsteady Euler Airfoil Solutions Using Unstructured Dynamic Meshes. AIAA Journal, 1990, 28(8): 1381-1388.
- [10] H. Xia and N. Qin. Dynamic grid and unsteady boundary conditions for synthetic jet flow. AIAA Paper 2005-106; Jan. 2005.